

Green Nature and Reducing of Air Pollution with Vehicle Drag Coefficient Correction

¹Abdollah Khalesi Doost, ²Ali Mohammad Seif Zadeh Yazdi

^{1,2} Islamic Azad University Semnan Branch

¹a.khalesi@semnaniau.ac.ir; ²ali_msy18@yahoo.com

Abstract

Drag coefficient can be calculated by means of corridors in the air or through computer software. The use of computer software makes it possible to be in time and cost savings. Numerical methods for doing this can be the fluid flow by a two dimensional or three-dimensional model of the vehicle structure to be resolved. This paper has tried to use the FLUENT and the equations governing fluid flow combined with standard model solved by using the appropriate boundary conditions. The study results of the aerodynamic analysis observing the flow lines, Vortex generated around the car and the pressure distribution on the vehicle structure, the area that increases the drag force are identified, to present a method to reduce the drag force. Ideas presented in this paper are to channel air from the front bumper to rear bumper transfer and reduce the air pressure in front of the vehicle. Reduction in the vortex behind the car is covered in foam equipment. They have been calculated and compared with the drag coefficient in two channel modes and with the weather channel and it should be noted that the drag coefficient is reduced by 23%. So the drag coefficient reduces vehicle fuel consumption, consequently air pollution, and then resulted in green environment.

Keywords

Automotive Aerodynamics; Drag Coefficient; Fuel Consumption; Optimization; Numerical Analysis; Drag pressure; Reduce Air Pollution; Green Nature

Introduction

In modern trucks and automobiles, aerodynamic features are reflected in how these vehicles are in line with road load. Aerodynamic actions on vehicles create drag, lift, and lateral forces as well as noise and torque, rotary, and momentum. Such forces influence fuel consumption, controllability, and vibrations. Aerodynamic forces exerted on vehicles originate from two sources: force drag (or pressure) and viscous friction.

Although wind tunnel studies are extremely important, however, computational methods have been widely used in developed countries to simulate automotive bodies and the application of highly-efficient computers has reduced the required time and number of wind tunnel experiments. Despite all these advancements in computational methods, physical and geometric complexities involved in the problem still hinder correct and exact prediction of flows. For vehicles, we are faced with entirely turbulent 3D flows, flow separation, and reverse flow. Therefore, developing geometry, a mesh, and a problem-solving approach requires computational resources and reliable algorithms and tools. Limited volume algorithms applied to governing equations, are standard techniques commonly used for external flows in industrial applications. Required time and memory are two limiting factors in selection of the number of cells and type of turbulence model.

Numerical and Experimental Works for Reducing Drag Coefficient

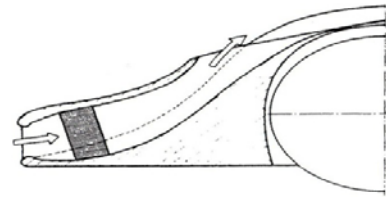
A large body of numerical and experimental studies has focused on reducing drag coefficient. Some of these works are in line with the purpose of the present study in connection with reducing the pressure exerted on the front part of vehicles, mitigating vortices behind vehicles, and eliminating drag caused by equipment used in the underbody. For years, many studies have attempted to analyze aerodynamic of vehicles. Research indicates that advancements in the area of vehicle aerodynamics largely focused on the shape of upper body for vehicles. In a modern vehicle, the overall shape contributes to 45% of drag while the shares of wheels, their arrangement, the bottom, and its design details, are 30% and 25%, respectively. Despite all studies, researches, and works on different

parts of vehicle, the drag problem is still a significant one, particularly for SUVs. In 1958, Hoerner pointed out that underbody was one of the most inevitable contributors to creation of drag forces which varied from 0.3 to for a vehicle with flat bottom to 0.6 for non-flat or open bottoms. In 1965, G. W. Carr developed a model to determine aerodynamic effects of underbody. In 1982, Holt suggested that airflow at underbody would become a major aerodynamic problem. In 1998, Cooper *et al* found that a major part of lift and drag forces were influenced by underbody. In 1998, 1999, and 2000, Skea *et al* and proposed some methods to examine underbody and to determine whether CFD was a proper analytical tool for this type of engineering problems. In 2000, Casella *et al* measured capabilities of CFD in studies on underbody. They found that CFD was a proper tool since its results were well consistent with the experimental results. In its latest 2003 model, Mitsubishi Lancer has particularly focused on underbody. In their 2004 study on comparison of analytical findings in connection with bottom parts of vehicles, Beheshti Borumand found that covering these parts in Pride vehicle can reduce drag coefficient by 14% since covering these parts using flat sheets results in a less turbulent flow under the vehicle and prevents airflow from contacting bottom parts. Thus, following those uneven parts in underbody, just as in front parts, it may create drag forces and therefore must be carefully taken into account.

FIG. 1 shows a type of vehicle in which radiator is designed with minimum impact on drag increase. Although air loses speed while passing through radiator, it is extremely effective in heat transfer.

In 1967, F. Bosnjakovic showed that air enters the channel with an initial velocity which is reduced when air meets radiator parts and enters areas of channel with larger cross sections. At the output, the velocity increases and reaches its initial value as channel cross section becomes smaller (a typical situation seen in jet nozzles). In 1980, G. Amato showed that such system creates a counter pressure (e.g. in reaction engines) and dissipated thermal energy compensates the loss in pressure. The situation is difficult one for in vitro simulation since air, space, and thermal sources are subject to some limitations.

FIG. 1 COOLING AIR DUCT FOR A FRONT-END RADIATOR. THE PURPOSEFUL AIR DUCT CONFIGURATION AND THE HIGH PRESSURE HEAD RESULT IN A HIGH COOLING EFFICIENCY



Vehicle Geometry

The vehicle used for aerodynamic analysis is a SUV one. Its high drag coefficient and sufficient space at underbody for placement of channel between the bars of the ladder chassis make it a good candidate for the purpose of this study. Ladder chassis is normally used in SUVs to create high resistance. In our design, ladder chassis is employed both to shield the channel and to provide a place where the channel can be installed.

Channel Design

The channel used in this design covers the space between the front and the rear bumpers. The channel height is adjustable to the chassis size in order to cover bottom parts as well as the void space in chassis. Although it is possible to calculate the channel height to achieve the optimized cross section, however, in some cases calculated channel height may be greater than chassis height. As it is impossible to reduce, limit, or displace underbody parts, the only option is to lower the channel with respect to chassis bottom which is not feasible in practice since the bottom of the channel may hit road bumps, creating noise, cracks, or fractures. The channel serves as a passage which conveys air to front chamber for combustion and cooling. The extra air may return to the main seat space.

Meshing

Given the geometrical complexities triangular cells can be used to create a mesh over the solution field. We used triangular meshes to save time in analysis of complex geometries. The cells are smaller in vicinity of the frame and become larger as we move away from the automobile frame. Smaller cells provide for finer accuracy and higher level of precision for repeated measurement of drag coefficient. FIGS. 2 and 3 illustrate the meshes used for this purpose.

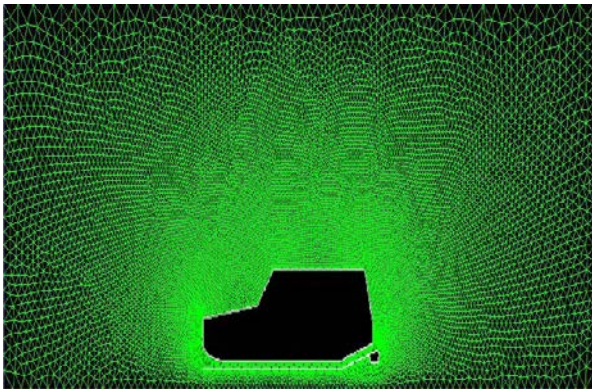


FIG. 2 MESH SIZE VEHICLE WITH DUCT-AIR

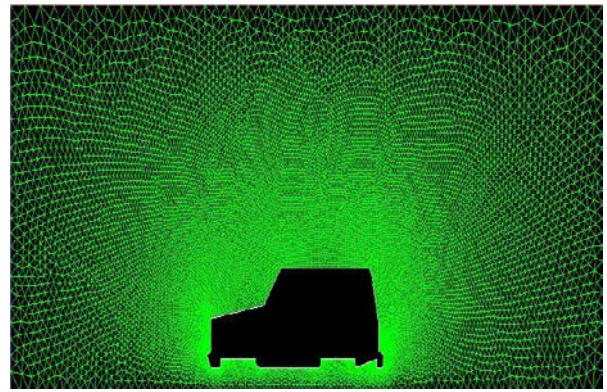


FIG. 3 MESH SIZE VEHICLE WITHOUT DUCT-AIR

Boundary Conditions

Given the various boundaries confronted in this problem, different boundary conditions must be taken into account. At the input, the velocity is 33.3 m/s and the turbulence intensity is known. The constant-pressure principle was applied to the output. In addition, wall functions were used to simulate no-slip condition for ground and solid surfaces. Air was considered an ideal gas with turbulent and compressible flow. Moreover, the small velocity of the vehicle compared to the speed of sound, a constant density of 1.225 kg/m³ at standard conditions was assumed for air. Since the flow was incompressible, viscosity was assumed to be 1.7894×10^{-5} N.s/m² for standard conditions.

Solving the Equations

By selecting a Cartesian coordinate system as the reference system, governing equations for a permanent incompressible turbulent three-dimensional flow may be expressed as follows: The equations include continuity equation, momentum equations for two directions in the coordinate system, and equations for the turbulence model (i.e. standard model). The second order equations were used for discretization of equations while SIMPLE and SIMPLEC algorithms were employed for pressure-velocity coupling.

Problem Solving Technique

In the first step, the vehicle was modeled through Gambit using a 2D model for normal conditions without incorporating the channel. Then, the meshes and boundary conditions in Section 6 were defined. Finally, the model was inserted into Fluent environment where equations and techniques referred

to in section 7 were implemented. Through iterations, the value of 0.7423 was obtained as the drag coefficient in the first step. This large value is caused by large non-flat surfaces at the front and rear parts, windshield, and breakpoints at the front area of the hood as well as the points connected to the windshield.

In the second step (the new geometry incorporating the channel), the model was moved into Fluent once the meshes and boundary conditions were defined in Gambit. Then the working conditions were implemented through a similar procedure described in the first step in order to determine the difference in drag coefficients obtained for the two steps. In the second step the equations were less convergent and eventually trapped in oscillations within a certain range. To tackle this issue, we gradually reduced the values of Density, Momentum, Turbulent Viscosity, and Energy step by step and then switched from SIMPLE to SIMPLEC and PISO where the corresponding values were increased. After each modification, "Iterate" was activated in Solver in order to find a possible solution and obtain the drag coefficient.

Since the issue was not resolved by this technique, we switched the Solver type from "Steady" to "Unsteady" and also changed "Absolute" to "Relative", and then activated "Iterate", giving 4118 as the solution. Then we restored the Solver's parameter "Unsteady" to "Steady" and obtained 4860 though iteration for the steady state. Finally, the parameter "Relative" was restored to its original state "Absolute" and "Iterate" was activated, giving 0.57156 as the final solution for drag coefficient at the second step (see FIG. 4. The steps were designed based on 10.

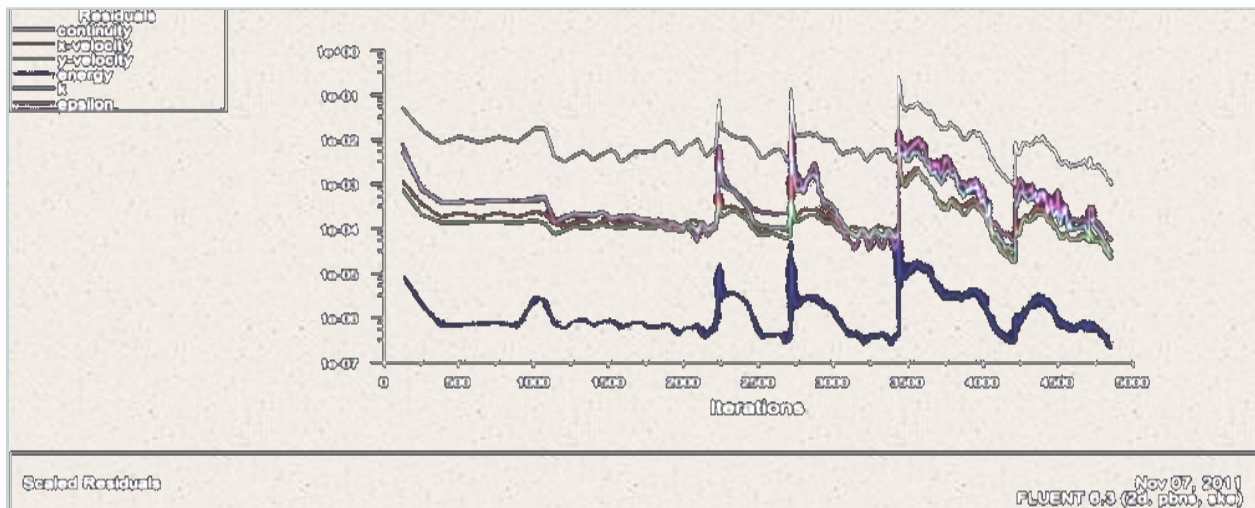


FIG. 4 BASED ON THE CONVERGENCE OF DIAGRAM THE SOLUTIONS LISTED

Analysis of Results

Drag coefficient was reduced by 23%; *i.e.* from 0.7423 to 0.57156. It was followed that by creating this type of channel under different types of vehicles, race cars, and particularly SUVs, one can significantly decrease total drag coefficient of the vehicles.

FIG. 5 and 6 as well as velocity vectors and contours (FIG. 7 and 8), indicated that flow has become smoother around the vehicle and there is smaller number of vortices behind the vehicle. In addition, the

high-speed area above the original (channel less) vehicle - that extends from the front to rear parts - has been reduced comparing with the channel-equipped vehicle. As seen in FIGS. 9 and 10 (pressure vectors) and FIGS. 11 and 12 (pressure contours) the pressure has been lowered in front of the windshield and front bumper, resulting in almost equal pressures at front and rear which, in turn, reduce drag forces. A comparison of the figures indicates that for the channel-equipped vehicle, the pressure at the upper body is equal to pressure at the underbody, creating a negative lift and improving controllability of the vehicle.

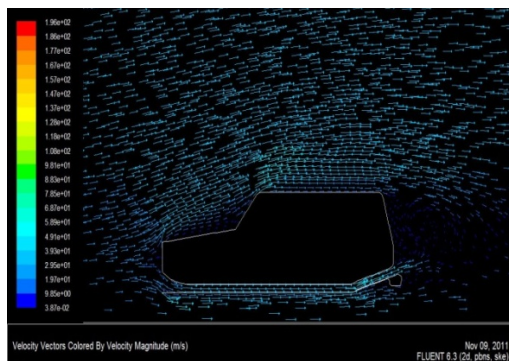


FIG. 5 VELOCITY VECTOR OF THE VEHICLE WITH DUCT

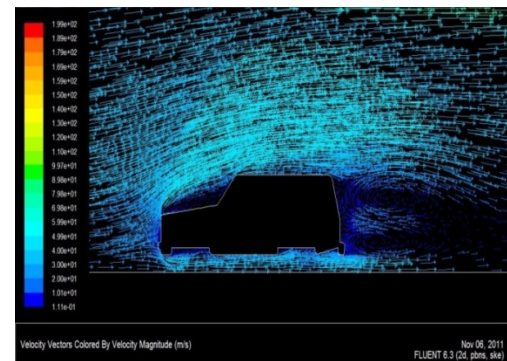


FIG. 6 VELOCITY VECTOR OF THE VEHICLE WITHOUT DUCT-AIR

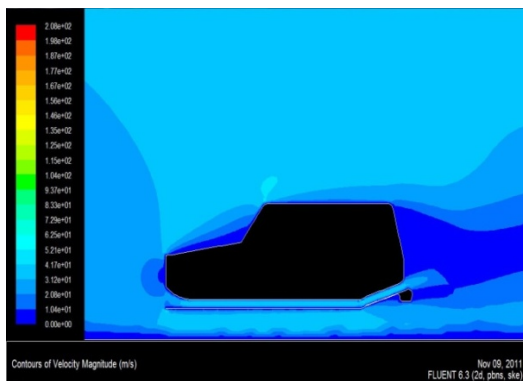


FIG. 7 VELOCITY CONTOURS OF THE VEHICLE WITH DUCT-AIR



FIG. 8 VELOCITY CONTOURS OF THE VEHICLE WITHOUT DUCT-AIR

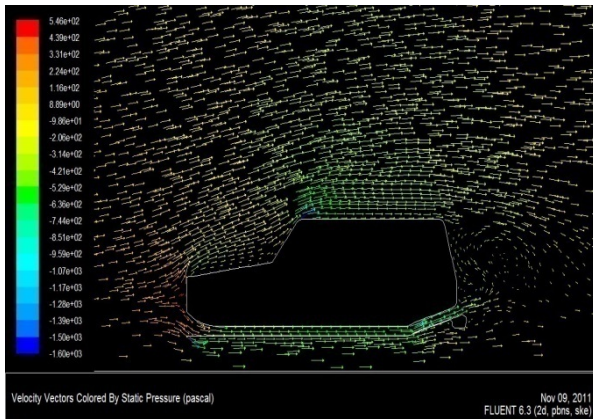


FIG. 9 PRESSURE VECTOR OF THE VEHICLE WITH DUCT-AIR

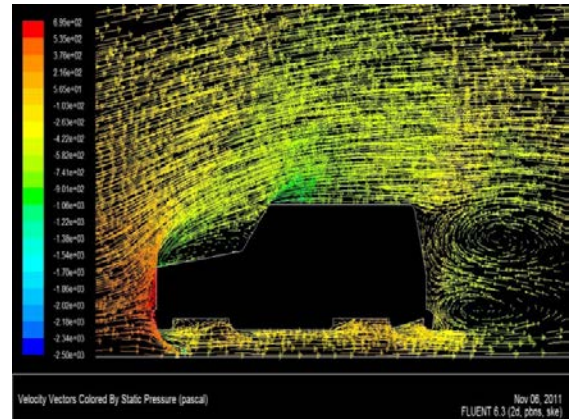


FIG. 10 PRESSURE VECTOR OF THE VEHICLE WITHOUT DUCT- AIR

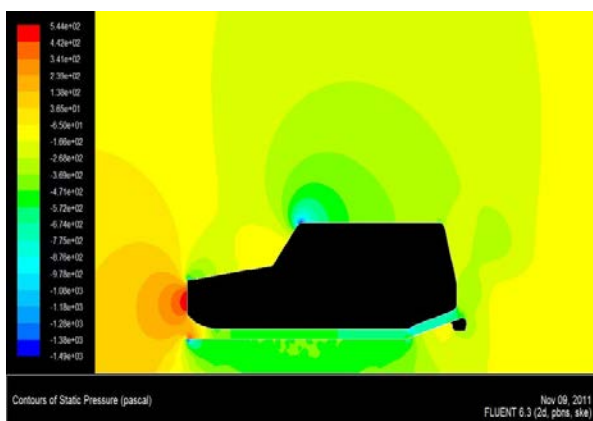


FIG. 11 PRESSURE CONTOURS OF THE VEHICLE WITH DUCT-AIR

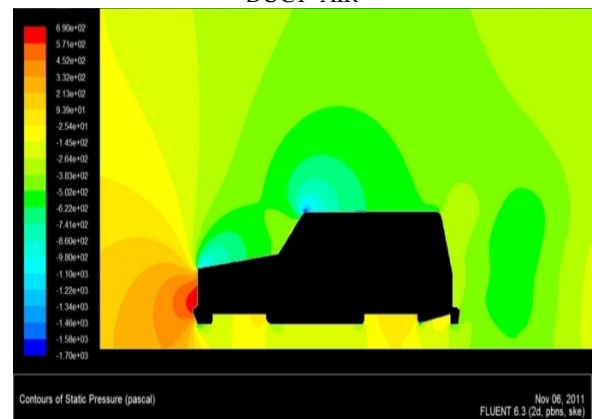


FIG. 12 PRESSURE CONTOURS OF THE VEHICLE WITHOUT DUCT-AIR

This study has two main goals:

- 1- Reduction in drag coefficient and almost equal air pressure at either sides of the vehicle caused by airflow from front to rear section
- 2- Reduction in drag coefficient attributable to bottom parts, through covering underbody with the air channel

Potential benefits of the design:

- (1) air pressure balance (reduced difference in pressure at front and rear) resulting in decreased drag forces and drag coefficient;
- (2) improvement in vehicle speed;
- (3) lower fuel consumption;
- (4) reduction in air pollution;
- (5) smaller engine size, and in turn, reduced weight

Conclusion

Idea presented in this paper is to channel air from the front bumper to rear bumper transfer and reduce the air pressure in front of the vehicle. Reduction in the vortex behind the car is covered in foam equipment.

The proposed design reduces drag coefficient by 23%, directly improving fuel consumption in vehicles. This remarkable save in fuel provides reduction in air pollution and brings about a green environment.

REFERENCES

- BEHESHTI BOROMAND, B., Evaluation of parameters affecting the aerodynamic optimization of car, master's thesis, Department of Aerospace Engineering, Amirkabir University of Technology, 2004.
- CARR, G. W., 'New MIRA Drag Reduction Prediction Method for Cars', Automotive Engineer, June/July, 1987.
- CASELLA, M., MIROLO, E., RIBALDONE, E., and SCANTAMBURLO, G., the Use of CFD Techniques in the Solution of Automotive Problems, JSAE Spring Convention, Paper 20005346, 2000.
- COOPER, K. R., BERTENYI, T., DUTIL, G., SYMS, J., and SOVRAN, G., "The Aerodynamic Performance of Underbody Diffusers", SAE 980030, 1998.

- Fluent Users Manual, Chapter 9, Turbulence Models, 1998.
- GILLESPIE, T., (Author), KAZEMI, R., ANSARI MOVAHED, M. M. (Translator). Fundamentals of Vehicle Dynamics, 2002.
- HOERNER, S. F., "Fluid Dynamic Drag. Practical Information on Aerodynamic Drag and Hydrodynamic Resistance", Published by author, MIDLAND PARK, N. J., 1958.
- HOLT, D. J., Underbody Aerodynamics: The Next Area of Refinement, SAE Journal, 1982.
- SEIF ZADEH, Y. A. M., Decrease Drag Coefficient of the Vehicle Patrol to Help Duct-Air, the letter Engineering, Faculty of Engineering, Islamic Azad University, Semnan, 2011.
- SKEA, A. F., and BULLEN, P. R., and QIAO, J., Review of Underbody Aerodynamics: Testing Techniques; Airflow Characteristics; CFD Contribution, Ford Technical Journal, 2000.
- SKEA, A. F., and BULLEN, P. R., and QIAO, J., Underbody Aerodynamics: Using CFD to Simulate the Air Flow Around a Rotating wheel of a Passenger Car, Auto tech, Birmingham, 1999.
- SLACK, M., Cyclonic Separator, QNET-CFD Application Challenge, <http://www.qnet-cfd.net>, Accessed on 8th August 2003.
- WOLF - HUNCHO, H (Edited by). Aerodynamics of Road Vehicles, 4th Edition, SAE International, 1998.